EXHIBIT 17

DyMESH ®

CATEGORY: SOFTWARE

DyMESH ®

DyMESH (Dynamic MEchanical SHell)

DyMESH is a computer model for 3-D dynamic simulation of motor vehicle crashes. DyMESH uses a 3-D vehicle mesh with mechanical properties as input and produces vehicle-fixed collision forces and moments as output. DyMESH is a breakthrough technology for design engineers and safety researchers who study collision mechanics and crashworthiness.

DyMESH employs methods from finite element technology for collision detection, and stress-strain relationships for force calculation. Whereas a finite element analysis requires several hours (sometimes

days), a simulation using DyMESH is complete in a few minutes.

Using the HVE simulation environment, vehicle deformation is visualized as it is calculated during collision or rollover simulations. Researchers can compare simulated damage against actual damage from staged collisions or real world crashes as a means to validate their simulation results. Because the vehicle mesh typically includes several thousand nodes, HVE displays the resulting damage with great resolution.

DyMESH is useful for all collision simulations, and is especially useful for underride, or any crash where three-dimensional collision dynamics are present. All types of vehicles (passenger car, truck, trailer, dolly, barrier) may be involved in any number of simultaneous collisions. Results from DyMESH agree favorably with theory, test and finite element results. Detailed validation results can be found in SAE 1999-01-0104, "The DyMESH Method for Three-Dimensional Multi-Vehicle Collision Simulation" and SAE 2000-01-0844, "Validation of DyMESH for Vehicles vs. Barrier Collisions" and also SAE 2004-01-1207, "Validation of the SIMON Model for Vehicle Handling and Collision Simulation - Comparison of Results with Experiments and Other Models".

DyMESH calculates the forces and moments between interacting vehicle meshes. Concepts used in traditional finite element analysis are employed to

detect the penetration of a slave node into a master surface. Once penetration is detected, the pushback direction is determined, and the penetrating node is restored consistent with the kinematic constraint between the two surfaces. Forces (Fx, Fy, Fz) acting on a node are then calculated according to the node's mechanical properties (stress-strain relationship). Restitution is modeled as unloading of the slave nodes. Unloading begins when the node deformation rate reaches zero. Force calculations are performed for every interacting master surface and slave node.

574-E Ritchie Highway, Severna Park, MD 21146

503-644-4500 |